



Element Library Manual

Bořek Patzák

Czech Technical University
Faculty of Civil Engineering
Department of Structural Mechanics
Thákurova 7, 166 29 Prague, Czech Republic

July 28, 2009

Contents

| | | |
|----------|---|-----------|
| 1 | Introduction | 3 |
| 2 | Elements for Structural Analysis (SM Module) | 3 |
| 2.1 | Truss Elements | 3 |
| 2.1.1 | Truss 1D element | 3 |
| 2.1.2 | Truss 2D element | 3 |
| 2.2 | Beam Elements | 4 |
| 2.2.1 | Beam2d element | 4 |
| 2.2.2 | Beam3d element | 5 |
| 2.3 | Plane Stress Elements | 6 |
| 2.3.1 | PlaneStress2d | 6 |
| 2.3.2 | QPlaneStress2d | 7 |
| 2.3.3 | TrPlaneStress2d | 8 |
| 2.3.4 | QTrPlStr | 8 |
| 2.3.5 | TrPlaneStrRot | 9 |
| 2.4 | Plane Strain Elements | 9 |
| 2.4.1 | Quad1PlaneStrain | 9 |
| 2.4.2 | TrplaneStrain | 10 |
| 2.5 | Plate&Shell Elements | 11 |
| 2.5.1 | CCT Element | 11 |
| 2.5.2 | RerShell Element | 11 |
| 2.6 | Axisymmetric Elements | 11 |
| 2.6.1 | Axisymm3d element | 11 |
| 2.6.2 | Q4axisymm element | 12 |
| 2.6.3 | L4axisymm element | 12 |
| 2.7 | 3d Continuum Elements | 12 |
| 2.7.1 | LSpace element | 12 |
| 2.7.2 | LSpaceBB element | 13 |
| 2.7.3 | QSpace element | 13 |
| 2.7.4 | LTRSpace element | 14 |
| 2.8 | Interface elements | 15 |
| 2.8.1 | Interface2dquad element | 15 |
| 2.8.2 | Interface3dtrlin element | 16 |
| 2.8.3 | Interface1d element | 16 |
| 3 | Elements for Transport problems (TM Module) | 17 |
| 3.1 | 2D Elements | 17 |
| 3.1.1 | Quad1ht element | 17 |
| 3.1.2 | Quad1hmt element | 18 |
| 3.1.3 | Tr1ht element | 18 |
| 3.2 | Axisymmetric Elements | 18 |
| 3.2.1 | Quadaxisym1ht element | 18 |
| 3.2.2 | Traxisym1ht element | 18 |
| 3.3 | 3D Elements | 19 |
| 3.3.1 | Brick1ht element | 19 |
| 3.3.2 | Brick1hmt element | 19 |

| | | |
|----------|---|-----------|
| 4 | Elements for Fluid Dynamics problems | 20 |
| | (FM Module) | |
| 4.1 | 2D CBS Elements | 20 |
| 4.1.1 | Tr1CBS element | 20 |
| 4.2 | 2D SUPG/PSGP Elements | 21 |
| 4.2.1 | Tr1SUPG element | 21 |
| 4.2.2 | Tr1SUPGAXI element | 21 |

List of Figures

| | | |
|----|--|----|
| 1 | Truss2d element in (x,z) plane. | 4 |
| 2 | Beam2d element. Definition of local c.s.(a) and definition of local end forces and local element dofs (b). | 4 |
| 3 | Beam3d element. Definition of local c.s., local end forces and local element dofs numbering. | 5 |
| 4 | PlaneStress2d element. Node numbering, Side numbering and definition of local edge c.s.(a). | 6 |
| 5 | QPlaneStress element - node numbering. | 7 |
| 6 | TrPlaneStress element - node and side numbering. | 8 |
| 7 | TrPlaneStress element - node and side numbering. | 9 |
| 8 | Quad1PlaneStrain element. Node numbering, Side numbering and definition of local edge c.s.(a). | 10 |
| 9 | TrplaneStrain element - node and side numbering. | 10 |
| 10 | lspace element (Node numbers in black, side numbers in blue, and surface numbers in red). | 13 |
| 11 | qspace element. | 14 |
| 12 | Linear tetrahedra element. Definition and node numbering convention | 14 |
| 13 | 2D interface element with quadratic interpolation. Definition and node numbering convention | 15 |
| 14 | 3D interface element with linear interpolation. Definition and node numbering convention | 16 |
| 15 | Quad1ht element. Node numbering, Side numbering and definition of local edge c.s.(a). | 17 |
| 16 | TrPlaneStress element - node and side numbering. | 18 |
| 17 | brick element (Node numbers in black, side numbers in blue, and surface numbers in red). | 19 |
| 18 | Tr1CBS element. Node numbering, Side numbering and definition of local edge c.s.(a). | 20 |
| 19 | Tr1SUPG element. Node numbering, Side numbering and definition of local edge c.s.(a). | 21 |
| 20 | Tr1SUPGAXi element. Node numbering, Side numbering and definition of local edge c.s.(a). | 22 |

1 Introduction

In this manual the detailed description of available elements is given. The actual availability of particular elements depends on OOFEM configuration. Elements are specified using element records, which are part of oofem input file. The general format of element record is described in OOFEM input manual.

Every element is described in a separate section. The section includes the “element keyword”, which determines the element type in element record, approximation and interpolation characteristics, required cross section properties (which are summarized in “CS properties” part), and a summary of element features. The “Load” section contains useful information about the types of loadings supported by particular elements.

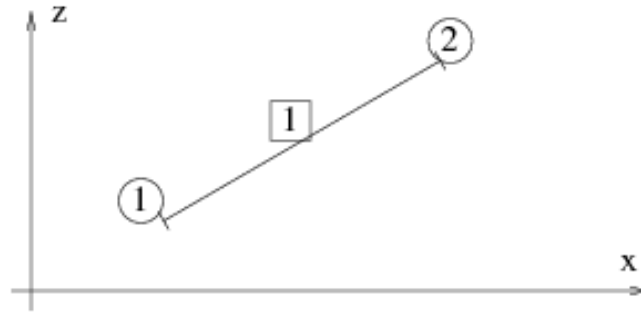


Figure 1: Truss2d element in (x,z) plane.

2 Elements for Structural Analysis (SM Module)

2.1 Truss Elements

2.1.1 Truss 1D element

Represents linear isoparametric truss element in 1D. The elements are assumed to be located along the x-axis. Requires cross section area to be specified.

Keyword: `truss1d`

Parameters: none.

Unknowns: Single dof (u-displacement) is required in each node .

Approximation: Linear approximation of displacement and geometry.

Integration: Exact.

Features: Full dynamic analysis support, Full nonlocal constitutive support, Adaptivity support

CS properties: Area is required.

Loads: Body loads are supported. Boundary loads are not supported in current implementation.

Status: Reliable

2.1.2 Truss 2D element

Two node linear isoparametric truss element for 2D analysis. The element geometry can be specified in (x,z) , (x,y) , or (y,z) plane.

Keyword: `truss2d`

Parameters: [`cs(in)` #]

Unknowns: Two dofs representing displacements in definition plane are required in each node. The element can be used in different planes, default definition plane is (x,z) . The parameter `cs` can be used to change default definition plane. The supported values of parameter `cs` are following: 0 for (x,z) plane (default), 1 for (x,y) plane, and 3 for (y,z) plane.

Approximation: Linear approximation of displacements and geometry.

Integration: Exact.

Features: Full dynamic analysis support. Full nonlocal constitutive support.

CS properties: cross section area should be provided.

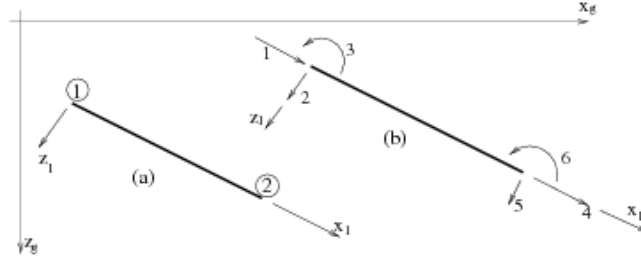


Figure 2: Beam2d element. Definition of local c.s.(a) and definition of local end forces and local element dofs (b).

Loads: Edge loads are supported, Edge number should be equal to 1.

Status: Reliable

2.2 Beam Elements

2.2.1 Beam2d element

Beam element for 2D analysis, based on Timoshenko hypothesis. Structure should be defined in x,z plane. The internal condensation of arbitrary DOF is supported and is performed in local coordinate system. On output, the local end displacement and local end forces are printed.

Keyword: beam2d

Parameters: [dofstocondense_(ia) #]

The dofstocondense parameter allows to specify local element dofs that will be condensed. The numbering of local element dofs is shown in fig. 2. The size of this array should be equal to number of local element dofs (6) and nonzero value indicates the corresponding dof will be condensed.

Unknowns: Three dofs (u-displacement, w-displacement, y-rotation) are required in each node.

Approximation: Cubic approximations of lateral displacement and rotation are used. For longitudinal displacement the linear one is assumed.

Integration: Exact.

Features: Full dynamic analysis support. Linear stability analysis support.

CS properties: Area, inertia moment along y-axis and area shear correction factor should be specified.

Loads: Constant and linear edge loads are supported, shear influence is taken into account. Edge number should be equal to 1. Temperature load is supported, the first coefficient of temperature load represent mid-plane temperature change, the second one represent difference between temperature change of local $z+$ and local $z-$ surfaces of beam (in local coordinate system). Temperature load require that the “thick” property of cross section model is defined.

Status: Reliable.

2.2.2 Beam3d element

Beam element for 3D **linear** analysis, based on Timoshenko hypothesis. The internal condensation of arbitrary DOF is supported and is performed in local coordinate system. On output, the local end-displacement and local end-forces

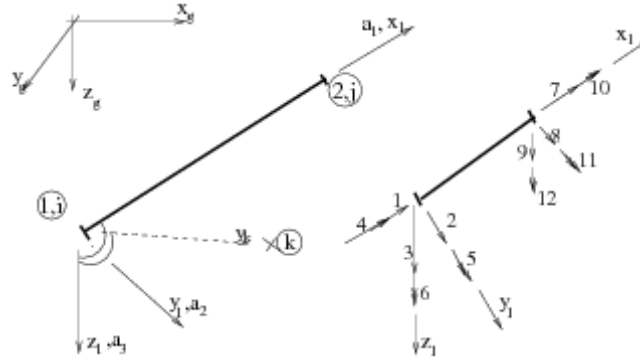


Figure 3: Beam3d element. Definition of local c.s., local end forces and local element dofs numbering.

are printed. Requires the local coordinate system to be chosen according to main central axes of inertia. Local element coordinate system is determined by the following rules:

1. let first element node has following coordinates (x_i, y_i, z_i) and the second one (x_j, y_j, z_j) ,
2. direction vector of local x-axis is then $\mathbf{a}_1 = (x_j - x_i, y_j - y_i, z_j - z_i)$,
3. local y-axis direction vector lies in plane defined by local x-axis direction vector (\mathbf{a}_1) and given point (k-node with coordinates (x_k, y_k, z_k)) - so called reference node,
4. local z-axis is then determined as vector product of local x-axis direction vector (\mathbf{a}_1) by vector $(x_k - x_i, y_k - y_i, z_k - z_i)$,
5. local y-axis is then determined as vector product of local z-axis direction vector by local x-axis direction vector.

Keyword: beam3d

Parameters: `refnode`_(in) # [`dofstocondense`_(ia) #]

The `refnode` parameter determines the reference node. It determines the local coordinate system of beam element. The `dofstocondense` parameter allows to specify local element dofs that will be condensed. The numbering of local element dofs is shown in fig. 3. The size of this array should be equal to number of local element dofs (12) and nonzero value indicates the corresponding dof will be condensed.

Unknowns: Six dofs (u,v,w-displacements and x,y,z-rotations) are required in each node.

Approximation: Cubic approximations of lateral displacement and rotation (along local y,z axes) are used. For longitudinal displacement and the rotation along local x-axis (torsion) the linear approximations are assumed.

Integration: Exact.

Features: Full dynamic analysis support. Linear stability analysis support.

CS properties: Area, inertia moment along y and z axis, torsion inertia moment and cross section area shear correction factor are required. These cross

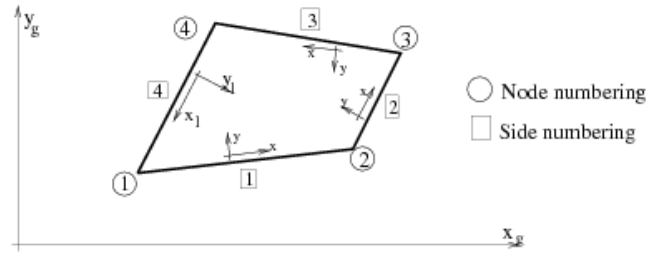


Figure 4: PlaneStress2d element. Node numbering, Side numbering and definition of local edge c.s.(a).

section properties are assumed to be defined in local coordinate system of element.

Loads: Constant and linear edge loads are supported. Edge number should be equal to 1. Temperature load is supported, the first coefficient of temperature load represent mid-plane temperature change, the second one represent difference between temperature change of local $z+$ surface and local $z-$ surface surface of beam and the third one represent difference between temperature change of local $y+$ surface and local $y-$ surface of beam. Requires the “thick” (measured in direction of local z axis) and “width” (measured in direction of local y axis) cross section model properties to be defined.

Status: Stable, various loadings require further testing.

2.3 Plane Stress Elements

2.3.1 PlaneStress2d

Represents isoparametric four-node quadrilateral plane-stress finite element. Each node has 2 degrees of freedom. Structure should be defined in x,y plane. The nodes should be numbered anti-clockwise (positive rotation around z -axis).

Keyword: `planestress2d`

Parameters: $[NIP_{(in)} \#]$

Unknowns: Two dofs (u-displacement, v-displacement) are required in each node.

Approximation: Linear approximation of displacement and geometry.

Integration: Integration of membrane strain terms using gauss integration formula in 4 (the default), 9, or 16 integration points. The default number of integration point used can be overloaded using `NIP` parameter. Reduced integration for shear terms is employed. Shear terms are always integrated using 1 point integration rule.

Features: Nonlocal constitutive support, Geometric nonlinearity support.

CS properties: Thickness.

Loads: Body loads are supported. Boundary loads are supported and computed using numerical integration. The side numbering is following. Each i -th element side begins in i -th element node and ends on next element node ($i+1$ -th node or 1-st node, in the case of side number 4). The local positive edge x -axis coincides with side direction, the positive local edge y -axis is rotated 90 degrees anti-clockwise (see fig. (4)).

Status: Reliable.

2.3.2 QPlaneStress2d

Implementation of quadratic isoparametric eight-node quadrilateral plane-stress finite element. Each node has 2 degrees of freedom. The node numbering is anti-clockwise and is explained in fig. (5).

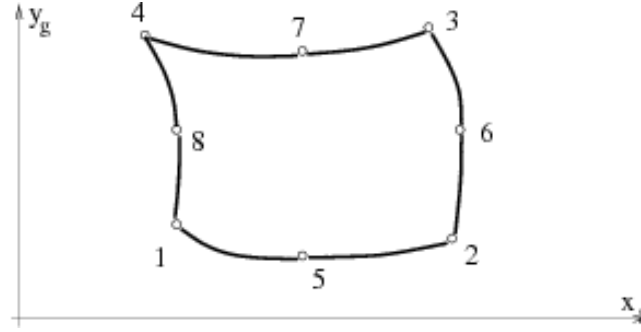


Figure 5: QPlaneStress element - node numbering.

Keyword: qplanestress2d

Parameters: [NIP_(in) #]

Unknowns: Two dofs (u-displacement, v-displacement) are required in each node.

Approximation: Quadratic approximation of displacement and geometry.

Integration: Full integration using gauss integration formula in 4 (the default), 9, or 16 integration points. The default number of integration point used can be overloaded using NIP parameter.

Features: Adaptivity support.

CS properties: Thickness.

Loads: Body loads are supported. Boundary loads are not supported in current implementation.

Status: Stable.

2.3.3 TrPlaneStress2d

Implements an triangular three-node constant strain plane-stress finite element. Each node has 2 degrees of freedom. The node numbering is anti-clockwise

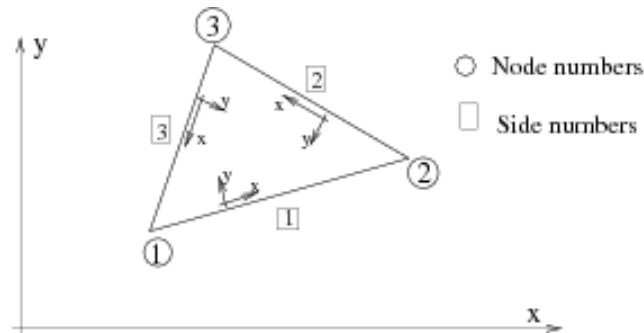


Figure 6: TrPlaneStress element - node and side numbering.

Keyword: trplanestress2d

Parameters: none.

Unknowns: Two dofs (u-displacement, v-displacement) are required in each node.

Approximation: Linear approximation of displacement and geometry.

Integration: Integration of membrane strain terms using one point gauss integration formula.

Features: Nonlocal constitutive support, Edge load support, Geometric non-linearity support, Adaptivity support

CS properties: Thickness.

Loads: Body loads are supported. Boundary loads are supported and are computed using numerical integration. The side numbering is following. Each i-th element side begins in i-th element node and ends on next element node (i+1-th node or 1-st node, in the case of side number 3). The local positive edge x-axis coincides with side direction, the positive local edge y-axis is rotated 90 degrees anti-clockwise (see fig. (6)).

Status: Reliable.

2.3.4 QTrPlStr

Implementation of quadratic six-node plane-stress finite element. Each node has 2 degrees of freedom. Node numbering is anti-clockwise and is shown in fig. (7).

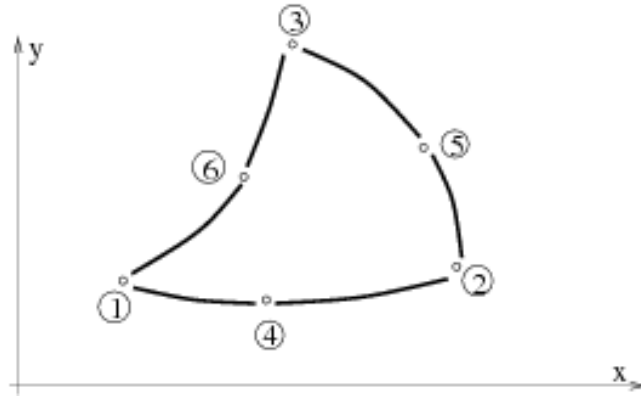


Figure 7: TrPlaneStress element - node and side numbering.

Keyword: qtrplstr

Parameters: [NIP_(in) #]

Unknowns: Two dofs (u-displacement, v-displacement) are required in each node.

Approximation: Quadratic approximation of displacement and geometry.

Integration: Full integration using gauss integration formula in 4 points (the default) or in 7 points (using NIP parameter).

Features: Adaptivity support (error indicator).

CS properties: Thickness.

Loads:

Status:

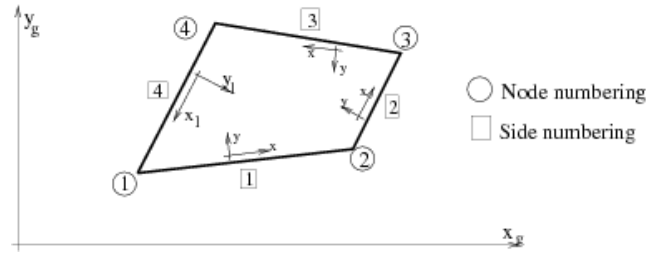


Figure 8: Quad1PlaneStrain element. Node numbering, Side numbering and definition of local edge c.s.(a).

2.3.5 TrPlaneStrRot

Implementation of triangular three-node plane-stress finite element with independent rotation field. Each node has 3 degrees of freedom.

Keyword: `trplanestrrot`

Parameters: $[NIP_{(in)} \#] [NIPRot_{(in)} \#]$

Unknowns: Three dofs (u-displacement, v-displacement, z-rotation) are required in each node.

Integration: Integration of membrane strain terms using gauss integration formula in 4 points (default) or using 1 or 7 points (using NIP parameter). Integration of strains associated with rotational field integration using 1 point is default (4 and 7 points rules can be specified using NIPRot parameter).

CS properties: Thickness.

Features:

Status:

2.4 Plane Strain Elements

2.4.1 Quad1PlaneStrain

Represents isoparametric four-node quadrilateral plane-strain finite element. Each node has 2 degrees of freedom. Structure should be defined in x,y plane. The nodes should be numbered anti-clockwise (positive rotation around z-axis).

Keyword: `quad1planestrain`

Parameters: $[NIP_{(in)} \#]$

Unknowns: Two dofs (u-displacement, v-displacement) are required in each node.

Approximation: Linear approximation of displacement and geometry.

Integration: Integration of membrane strain terms using gauss integration formula in 4 (the default), 9, or 16 integration points. The default number of integration point used can be overloaded using NIP parameter. Reduced integration for shear terms is employed. Shear terms are always integrated using 1 point integration rule.

Features: Nonlocal constitutive support, Adaptivity support.

CS properties: Thickness.

Loads: Body loads are supported. Boundary loads are supported and computed using numerical integration. The side numbering is following. Each i-th element side begins in i-th element node and ends on next element node (i+1-th node

or 1-st node, in the case of side number 4). The local positive edge x-axis coincides with side direction, the positive local edge y-axis is rotated 90 degrees anti-clockwise (see fig. (8)).

Status: Reliable.

2.4.2 TrplaneStrain

Implements an triangular three-node constant strain plane-strain finite element. Each node has 2 degrees of freedom. The node numbering is anti-clockwise

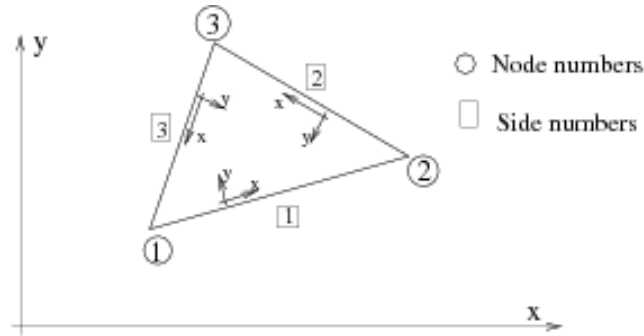


Figure 9: TrplaneStrain element - node and side numbering.

Keyword: trplanestrain

Parameters: none.

Unknowns: Two dofs (u-displacement, v-displacement) are required in each node.

Approximation: Linear approximation of displacement and geometry.

Integration: Integration of membrane strain terms using one point gauss integration formula.

Features: Nonlocal constitutive support. Edge load support, Adaptivity support.

CS properties: Thickness.

Loads: Body loads are supported. Boundary loads are supported and are computed using numerical integration. The side numbering is following. Each i-th element side begins in i-th element node and ends on next element node (i+1-th node or 1-st node, in the case of side number 3). The local positive edge x-axis coincides with side direction, the positive local edge y-axis is rotated 90 degrees anti-clockwise (see fig. (9)).

Status: Reliable.

2.5 Plate&Shell Elements

2.5.1 CCT Element

Implementation of constant curvature triangular element for plate analysis. Formulation based on Mindlin hypothesis. The structure should be defined in x,y plane. The nodes should be numbered anti-clockwise (positive rotation around z-axis).

Keyword: cctplate

Parameters: none.

Unknowns: Three dofs (w-displacement, u and v - rotation) are required in each node.

Integration: Integration of all terms using one point formula.

Features: Layered cross section support.

Loads: Body loads are supported. Boundary loads are not supported now.

Status: Reliable.

2.5.2 RerShell Element

Combination of CCT plate element (Mindlin hypothesis) with triangular plane stress element for membrane behavior. The element curvature can be specified. Although element requires generally six dofs per node, no stiffness to local rotation along z-axis (rotation around element normal) is supplied.

Keyword: rershell

Parameters: none.

Integration: Integration of all terms using one point formula.

Features: Layered cross section support.

Loads: Body loads are supported. Boundary loads are not supported now.

Status: Reliable.

2.6 Axisymmetric Elements

2.6.1 Axisymm3d element

Implementation of triangular three-node finite element for axisymmetric continuum. Each node has 2 degrees of freedom.

Keyword: axisymm3d

Parameters: [NIP_(in) #] [NIPfish_(in) #]

Unknowns: Two dofs (u-displacement, v-displacement) are required in each node.

Approximation: Linear approximation of displacement and geometry.

Integration: The integration of ε_x and ε_y strains can be altered using NIP parameter (possible completions are 1 (default), 4 and 7 point integration rule). The remaining strain components (ε_ϕ and γ_{rz}) can be integrated using 1 (default), 4 and 7 integration point formulae (NIPfish parameter).

Features: None.

Loads: No boundary and body loads are supported.

Status:

2.6.2 Q4axisymm element

Implementation of quadratic isoparametric eight-node quadrilateral - finite element for axisymmetric 3d continuum. Each node has 2 degrees of freedom.

Keyword: q4axisymm

Parameters: [NIP_(in) #] [NIPfish_(in) #]

Unknowns: Two dofs (u-displacement, v-displacement) are required in each node.

Approximation: Quadratic approximation of displacements and geometry.

Integration: The integration of ε_x and ε_y strains can be altered using NIP

parameter (possible completions are 1 (default), 4, 9 and 16 point integration rule). The remaining strain components (ε_ϕ and γ_{rz}) can be integrated using 1 (default), 4, 9 and 16 integration point formulae (NIPfish parameter).

Features: None.

Loads: No boundary and body loads are supported.

Status:

2.6.3 L4axisymm element

Implementation of isoparametric four-node quadrilateral axisymmetric finite element. Each node has 2 degrees of freedom.

Keyword: l4axisymm

Parameters: [NIP_(in) #]

Unknowns: Two dofs (u-displacement, v-displacement) are required in each node.

Approximation: Linear approximation of displacements and geometry.

Integration: The integration of ε_x and ε_y strains can be altered using NIP parameter (possible completions are 1 (default), 4, 9 and 16 point integration rule). The remaining strain components (ε_ϕ and γ_{rz}) are integrated using one point integration formula.

Features: None.

Loads: No boundary and body loads are supported.

Status:

2.7 3d Continuum Elements

2.7.1 LSpace element

Implementation of Linear 3d eight - node finite element. Each node has 3 degrees of freedom.

Keyword: lspace

Parameters: [NIP_(in) #]

Unknowns: Three dofs (u-displacement, v-displacement, w-displacement) are required in each node. **Approximation:** Linear approximation of displacements and geometry. **Integration:** Full integration of all strain components. NIP parameter (possible completions are 8 (default) and 27) allows to change the integration formula.

Features: Adaptivity support, Geometric nonlinearity support.

Loads:

Status:

2.7.2 LSpaceBB element

Implementation of 3d brick eight - node linear approximation element with selective integration of deviatoric and volumetric strain contributions (B-bar formulation) for incompressible problems. Features and description identical to conventional lspace element, see section 2.7.1.

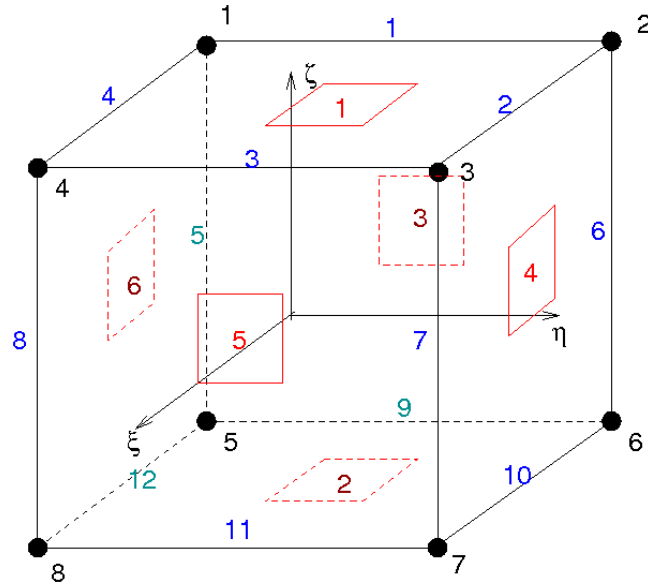


Figure 10: lspace element (Node numbers in black, side numbers in blue, and surface numbers in red).

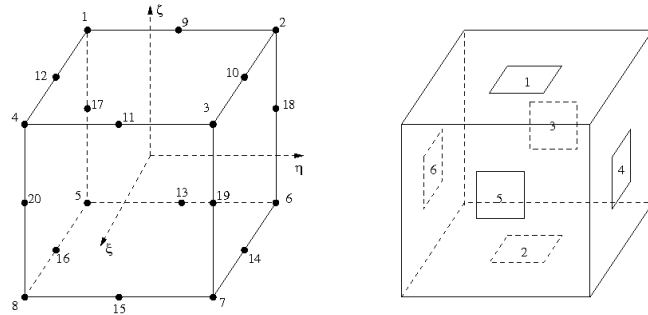


Figure 11: qspace element.

2.7.3 QSpace element

Implementation of quadratic 3d 20-node finite element. Each node has 3 degrees of freedom.

Keyword: qspace

Parameters: [$NIP_{(in)}$ #]

Unknowns: Three dofs (u-displacement, v-displacement, w-displacement) are required in each node.

Approximation: Quadratic approximation of displacements and geometry.

Integration: Full integration of all strain components. NIP parameter (possible completions are 8 (default) and 27) allows to change the integration formula.

Features: None.

Loads:

Status:

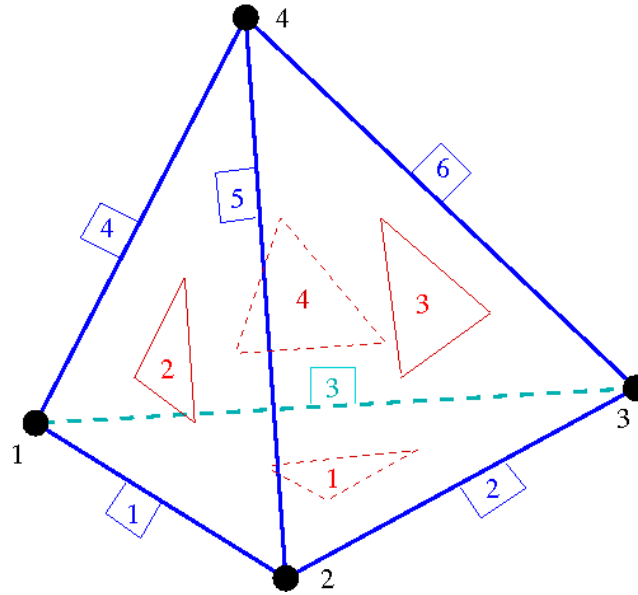


Figure 12: Linear tetrahedra element. Definition and node numbering convention

2.7.4 LTRSpace element

Implementation of tetrahedra four-node finite element. Each node has 3 degrees of freedom. Following node numbering convention is adopted (see also Fig. 12):

- Select a face that will contain the first three corners. The excluded corner will be the last one.
- Number these three corners in a counterclockwise sense when looking at the face from the excluded corner.

Keyword: LTRSpace

Parameters: None.

Unknowns: Three dofs (displacement in x,y, and z axis directions) are required in each node.

Approximation: Linear approximation of displacements and geometry using linear volume coordinates.

Integration: Full integration of all strain components using four point Gauss integration formula.

Features: Adaptivity support, Geometric nonlinearity support.

Loads: Surface and Edge loadings supported.

Status:

2.8 Interface elements

2.8.1 Interface2dquad element

Implementation of a two dimensional interface element with quadratic approximation of displacement field. Can be used to glue together two elements with quadratic displacement approximation along the shared edge. Note, that the

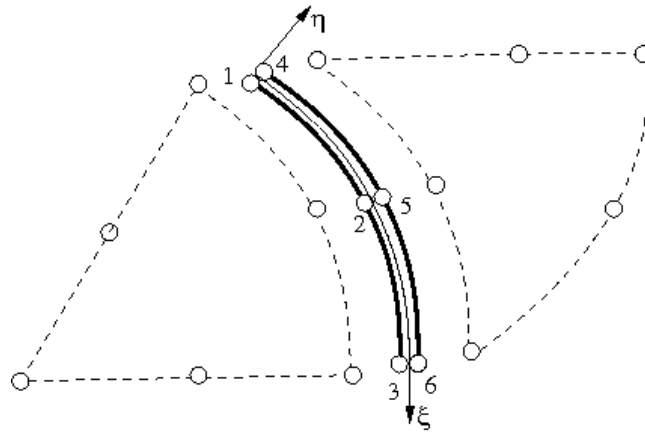


Figure 13: 2D interface element with quadratic interpolation. Definition and node numbering convention

nodes along the interface are doubled, each couple with identical coordinates. Nodes on the negative side are numbered first, followed by nodes on the positive part. Requires material model with `_2dInterface` support.

Keyword: `Interface2dquad`

Parameters: None.

Unknowns: Two dofs (u-displacement, v-displacement, w-displacement) are required in each node.

Approximation: quadratic approximation of displacements and geometry. **In-**

tegration: Full integration of all strain components using four point integration formula. **Features:** None.

Loads:

Status:

2.8.2 Interface3dtrlin element

Implementation of a three dimensional interface element with linear approximation of displacement field. Can be used to glue together two elements with linear displacement approximation along the shared tringular surface. Note, that the nodes along the interface are doubled, each couple with identical coordinates. Nodes on the negative surface are numbered first, followed by nodes on the positive part. The numbering of surface nodes on interface determines the positive normal (right hand rule). The surface in the direction of positive normal is the positive surface. Requires material model with `_3dInterface` support.

Keyword: `Interface3dtrlin`

Parameters: None.

Unknowns: Two dofs (u-displacement, v-displacement, w-displacement) are required in each node.

Approximation: Linear approximation of displacements and geometry.

Integration: Full integration of all components using one point integration formula. **Features:** None.

Loads:

Status:

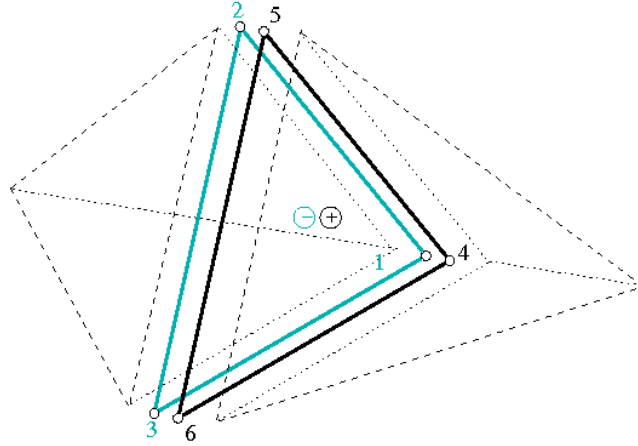


Figure 14: 3D interface element with linear interpolation. Definition and node numbering convention

2.8.3 Interface1d element

Implementation of one dimensional (slip) interface element. This element can connect two separate nodes by specifying the one-dimensional slip law, that determines the force acting between these nodes depending on their relative displacement. This element can be applied in 1D, 2D, and 3D settings (default).

Keyword: Interface1d

Parameters: refnode_(in) #

The **refnode** determines the reference node, which is used to specify a reference direction (the direction vector is obtained by subtracting coordinates of first node from reference node). The magnitude of slip is then obtained as relative displacement vector of two element nodes projected to reference direction. As a consequence, this will lead to linearized geometrical equations, where slip is always related to the original (undeformed) configuration. Element requires material model with `_1dInterface` support. **Unknowns:** One, two, or three DOFs (u-displacement, v-displacement, w-displacement) are required in each node, according to element mode.

Approximation: None. **Integration:** None. **Features:** None.

Loads:

Status:

3 Elements for Transport problems (TM Module)

3.1 2D Elements

3.1.1 Quad1ht element

Represents isoparametric four-node quadrilateral finite element for heat transfer problems. Each node has 1 degrees of freedom. Problem should be defined in x,y plane. The cross section thickness property is requested from cross section

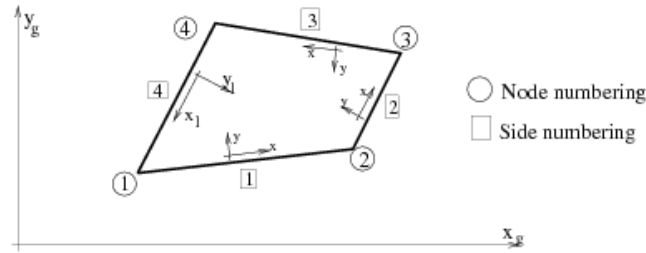


Figure 15: Quad1ht element. Node numbering, Side numbering and definition of local edge c.s.(a).

model. The nodes should be numbered anti-clockwise (positive rotation around z -axis).

Keyword: Quad1ht

Parameters: [$NIP_{(in)}$ #]

Unknowns: Single dof (T_f - temperature) is required in each node.

Approximation: Linear approximation of temperature.

Integration: Integration using gauss integration formula in 4 (the default), 9, or 16 integration points. The default number of integration point used can be overloaded using NIP parameter.

Loads: Body loads are supported. Boundary loads are supported and computed using numerical integration. The side numbering is following. Each i -th element side begins in i -th element node and ends on next element node ($i+1$ -th node or 1-st node, in the case of side number 4). The local positive edge x -axis coincides with side direction, the positive local edge y -axis is rotated 90 degrees anti-clockwise (see fig. (15)).

3.1.2 Quad1hmt element

Represents isoparametric four-node quadrilateral finite element for heat and mass (one constituent) transfer problems. Two dofs (T_f - temperature and C_1 - concentration) are required in each node. Linear approximation of temperature and mass concentration. Other features are similar to Quad1 element, see section 3.1.1.

3.1.3 Tr1ht element

Implements the linear triangular finite element for heat transfer problems. Each node has 1 degree of freedom. The cross section thickness property is requested from cross section model. The node numbering is anti-clockwise

Keyword: Tr1htfig

Parameters: none.

Unknowns: Single dof (T_f temperature) is required in each node.

Approximation: Linear approximation of temperature.

Integration: Integration using one point gauss integration formula.

Loads: Body loads are supported. Boundary loads are supported and are computed using numerical integration. The side numbering is following. Each i -th element side begins in i -th element node and ends on next element node ($i+1$ -th node or 1-st node, in the case of side number 3). The local positive edge

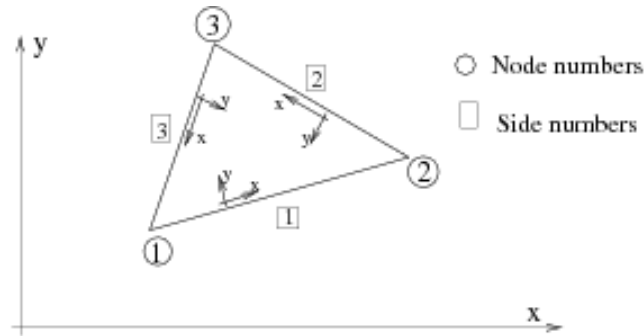


Figure 16: TrPlaneStress element - node and side numbering.

x-axis coincides with side direction, the positive local edge y-axis is rotated 90 degrees anti-clockwise (see fig. (16)).

3.2 Axisymmetric Elements

3.2.1 Quadaxisym1ht element

Isoparametric four-node quadrilateral finite element for axisymmetric heat transfer problems. The element description is similar to Quad1 element, see section 3.1.1.

3.2.2 Traxisym1ht element

Linear triangular finite element for axisymmetric heat transfer problems. The element description is similar to Tr1ht element, see section 3.1.3.

3.3 3D Elements

3.3.1 Brick1ht element

Represents isoparametric eight-node brick/hexahedron finite element for heat transfer problems. Each node has 1 degrees of freedom.

Keyword: Brick1ht

Parameters: [NIP_(in) #]

Unknowns: Single dof (T_f - temperature) is required in each node.

Approximation: Linear approximation of temperature.

Integration: Integration using gauss integration formula in 8 (the default), or 27 integration points. The default number of integration point used can be overloaded using NIP parameter.

Loads: Body loads are supported. Boundary loads are supported and computed using numerical integration. The side and surface numbering is shown in fig. (17)).

3.3.2 Brick1hmt element

Represents isoparametric eight-node quadrilateral finite element for heat and mass (one constituent) transfer problems. Two dofs (T_f - temperature and C₁

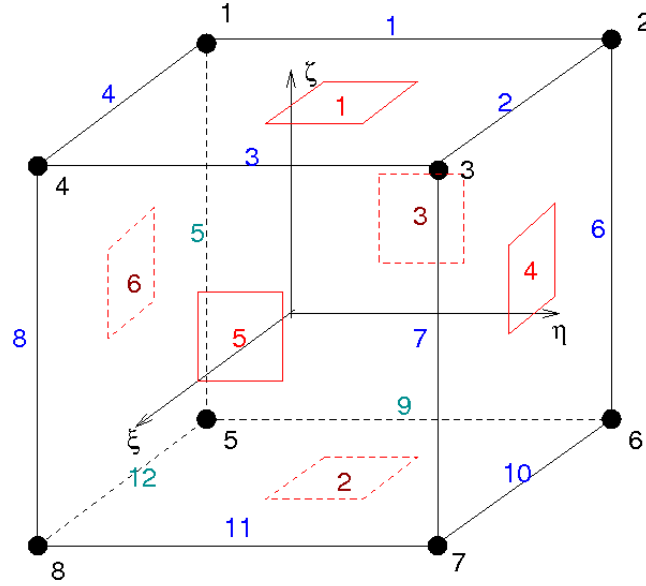


Figure 17: brick element (Node numbers in black, side numbers in blue, and surface numbers in red).

- concentration) are required in each node. Linear approximation of temperature and mass concentration. Other features are similar to Brick1 element, see section 3.3.1.

4 Elements for Fluid Dynamics problems (FM Module)

4.1 2D CBS Elements

4.1.1 Tr1CBS element

Represents the linear triangular finite element for transient incompressible flow analysis using cbs algorithm with equal order approximation of velocity and pressure fields. Each node has 3 degrees of freedoms (two components of velocity and pressure). The node numbering is anti-clockwise

Keyword: Tr1CBS

Parameters: [**bsides**_(ia) #] [**bcodes**_(ia) #]

Unknowns: Two velocity components (V_u and V_v) and pressure (P_f) are required in each node.

Boundary specification: Since the problem formulation requires to evaluate some boundary terms, the element boundary edges should be specified as well as the types of boundary conditions applied at these boundary edges. The boundary edges (their numbers) are specified using **bsides** array. The type of boundary condition(s) applied to corresponding boundary side is determined by **bcodes** array. The available/supported boundary codes are following: 1 for prescribed traction, 2 for prescribed normal velocity, 4 for prescribed tangential

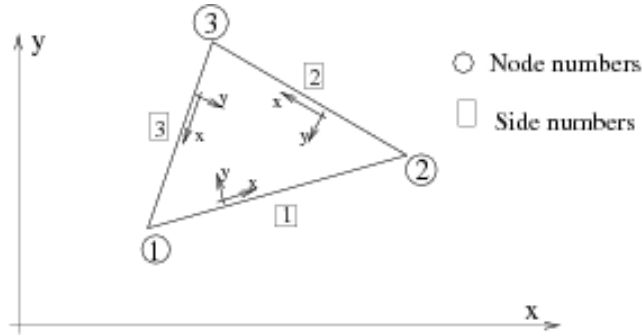


Figure 18: Tr1CBS element. Node numbering, Side numbering and definition of local edge c.s.(a).

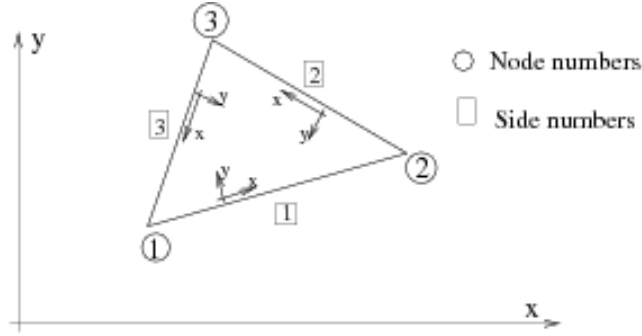


Figure 19: Tr1SUPG element. Node numbering, Side numbering and definition of local edge c.s.(a).

velocity, and 8 for prescribed pressure. If the element side is subjected to a combination of these fundamental types boundary conditions, the corresponding code is obtained by summing up the corresponding codes.

Approximation: Linear approximation of velocity and pressure fields.

Integration: exact

Loads: Constant boundary tractions are supported¹. Body loads representing the self-weight load are supported.

4.2 2D SUPG/PSGP Elements

4.2.1 Tr1SUPG element

Represents the linear triangular finite element for transient incompressible flow analysis using SUPG/PSPG stabilization with equal order approximation of velocity and pressure fields. Each node has 3 degrees of freedoms (two components of velocity and pressure). The node numbering is anti-clockwise

¹In CBS algorithm formulation the prescribed traction boundary condition leads indirectly to pressure boundary condition in nodes associated to loaded edge. Such boundary condition is represented by PrescribedTractionPressureBC. See section on boundary conditions in OOFEM input manual.

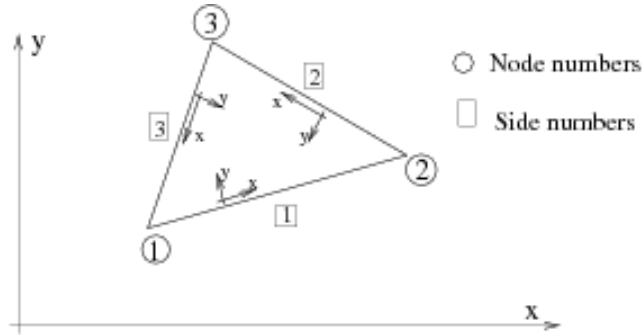


Figure 20: Tr1SUPGAxis element. Node numbering, Side numbering and definition of local edge c.s.(a).

Keyword: Tr1SUPG

Parameters: [$\text{vof}_{(rn)}$ #] [$\text{pvof}_{(rn)}$ #]

Unknowns: Two velocity components (V_u and V_v) and pressure (P_f) are required in each node.

Approximation: Linear approximation of velocity and pressure fields.

Integration: exact

Loads: Constant boundary tractions are supported. Body loads representing the self-weight load are supported.

Multi-fluid analysis: The element has support for solving problems with two immiscible fluids in a fixed spatial domain. In the present implementation, a VOF tracking algorithm is used to track the position of interface. An initial VOF fraction (volume fraction of reference fluid) can be specified using vof (default is zero). Element can also be marked as always filled with reference fluid (some form of source) using parameter pvof which specifies the permanent VOF value. In this case, the material model should be of type **Keyword:** `twofluidmat`, that supports modelling of two immiscible fluids.

4.2.2 Tr1SUPGAxis element

Represents the linear triangular finite element for transient incompressible flow analysis using SUPG/PSPG stabilization with equal order approximation of velocity and pressure fields in 2d-axisymmetric setting. Each node has 3 degrees of freedoms (two components of velocity and pressure). The y-axis is axis of rotational symmetry. The node numbering is anti-clockwise

Keyword: Tr1SUPGAxis

Parameters: [$\text{vof}_{(rn)}$ #] [$\text{pvof}_{(rn)}$ #]

Unknowns: Two velocity components (V_u and V_v) and pressure (P_f) are required in each node.

Approximation: Linear approximation of velocity and pressure fields.

Integration: Seven point Gauss integration.

Loads: Constant boundary tractions are supported. Body loads representing the self-weight load are supported.

Multi-fluid analysis: The element has support for solving problems with two immiscible fluids in a fixed spatial domain. In the present implementation, a VOF tracking algorithm is used to track the position of interface. An initial VOF

fraction (volume fraction of reference fluid) can be specified using `vof` (default is zero). Element can also be marked as always filled with reference fluid (some form of source) using parameter `pvo` which specifies the permanent VOF value. In this case, the material model should be of type **Keyword: twofluidmat**, that supports modelling of two immiscible fluids.